Semarak Journal of Thermal-Fluid Engineering

Journal homepage:
https://semarakilmu.com.my/journals/index.php/sjotfe/index
ISSN: 3030-6639

Flow Characteristics Effect on Different Blades Number of Radial Fan

Chan Kian Wui[1], Ishkrizat Taib[1,*], Wong Xin Yi[1]

[1] Department of Mechanical Engineering, Faculty of Mechanical Engineering and Manufacturing, Universiti Tun Hussein Onn Malaysia, 86400 Batu Pahat, Johor, Malaysia

ARTICLE INFO

Article history:
Received 24 July 2024
Received in revised form 7 August 2024
Accepted 21 August 2024
Available online 30 September 2024

Keywords:
Radial fan; ANSYS Fluent; mass flow rate; velocity; pressure

ABSTRACT

Radial fans are mainly used in industrial applications where working efficiency is of paramount importance. In order to examine the mass flow rate at the outlet for various configurations, three radial fan models with 12, 16, and 20 blades were modeled using ANSYS Fluent. The mass flow rate at the outlet of the 12-blade radial fan was determined to be 0.15897 kg/s. The mass flow rates for the 16-blade and 20-blade radial fans were 0.22092 kg/s and 0.22309 kg/s respectively, thus showing an increase in performance. The hypothesis of this study is therefore confirmed by the fact that as the number of blades increases, the spacing between them reduces ; hence, fluid flows at greater velocities, thereby enhancing the mass-flow rates. Verification and validation were established by comparing the results with those of other studies, which revealed a reasonable percentage error in each case. Furthermore, at the blade tips of the radial fan, the high static pressure, in conjunction with the radial velocity, reduces the vibration rates.

1. Introduction

Heavy industry, mining environments, and commercial buildings extensively use centrifugal fans. They facilitate the movement of air or gas particles toward the incoming fluid, thereby ensuring effective air distribution and maintaining air quality in specific settings [1-4]. Radial fans are known to produce significant pressure increases at lower flow rates than axial fans and are frequently used in heating, ventilation, and air conditioning (HVAC) systems [5]. These fans are crucial components in fluid flow operating systems. Radial fans consist of impellers powered by electric motors [6]. The rotation of the impeller directs air particles, allowing fluids to flow through ducts and overcome resistance, thus achieving the desired flow according to the design specifications [7].

A numerical study by Galloni *et al.,* [8] examined the performance and efficiency of turbomachinery based on the blade count. The results showed that as the number of blades increased, both the volumetric flow rate and dimensionless flow coefficient increased exponentially. An increase in the blade count reduces the slippage effect, thereby allowing the impeller to deliver more work to the fluid. Consequently, fans with more blades can move a greater airflow at the same

* *Corresponding author.*
E-mail address: iszat@uthm.edu.my

https://doi.org/10.37934/sjotfe.2.1.18

Semarak Journal of Thermal-Fluid Engineering
Volume 2, Issue 1 (2024) 1-8

counter pressure. Similarly, Aureliano *et al.,* [7] experimentally and numerically analyzed centrifugal ventilators and concluded that increasing the blade count enhances the output flow. More blades reduce the blade spacing by narrowing the fluid path and minimising centrifugal effects, thereby improving fluid flow and throughput.

The design of a radial fan impeller is vital for optimising performance. Impeller effectiveness is influenced by geometric parameters such as the blade count, shape, thickness, profile, and inlet/outlet diameters [9,10]. However, variations in the blade count affect the flow efficiency and flow characteristics, such as the flow rate, velocity, and pressure [9]. Numerical analyses using computational fluid dynamics (CFD) to solve nonlinear partial differential equations governing fluid flow, turbulence, and heat transfer. CFD is an advanced technique that is widely used in fluid flow research [11,12].

This project determines the flow characteristics near a radial fan's blades and explore the relationship between the blade count and fluid flow rate. It is hypothesised that increasing the blade count will improve the output flow. To achieve these goals, simulations of radial fans with 12, 16, and 20 blades were conducted using ANSYS Fluent for CFD analysis and SolidWorks for design. This study focused on varying the blade count while keeping the other parameters constant, and it validated the simulation results with the existing experimental data.

2. Methodology

2.1 Geometry of the Radial Fan

Radial fans were modelled using SolidWorks software, and the main components included the casing, impeller (featuring different blade counts), and inlet plate [13]. The overall dimensions of these models are shown in Figure 1. Three models were used in this project: radial fans with 12, 16, and 20 blades. Radial fans are generally divided into two main domains: volutes and rotors. This domain creation ensures an interface between the rotor and casing, where the flow is fully developed upon leaving the inlet and outlet. The blades and rotating region are defined as a rotating reference frame, whereas the fan casing is defined as a stationary frame [13,14].

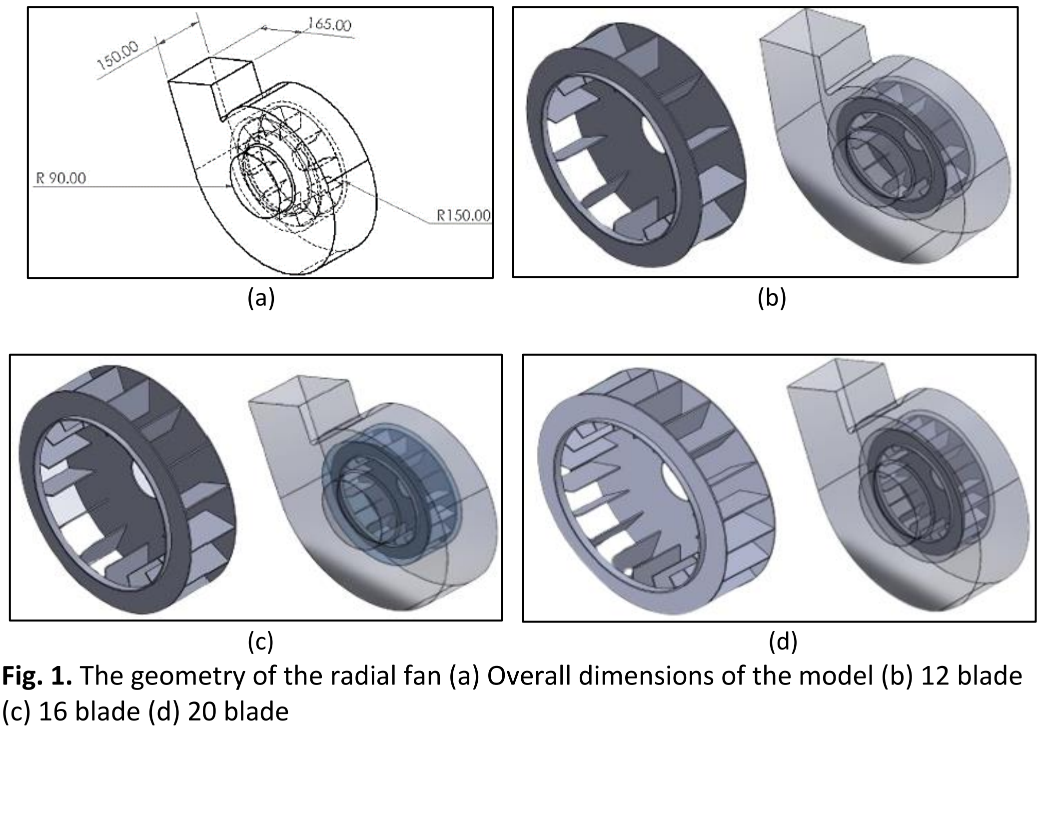

Fig. 1. The geometry of the radial fan (a) Overall dimensions of the model (b) 12 blade (c) 16 blade (d) 20 blade

2.2 Meshing of the Model

The meshing process discretizes the computational domain into numerous cells or elements [15]. Essentially, this process creates mathematical equations based on the fluid flow model, thereby enhancing the numerical analysis and evaluation. The number of elements and nodes affects the accuracy of the fluid flow solution in the finite element model [16]. High-quality meshing results in more accurate fluid flow solutions, although it requires more time. The general meshing method is a tetrahedron, as shown in Figure 2.

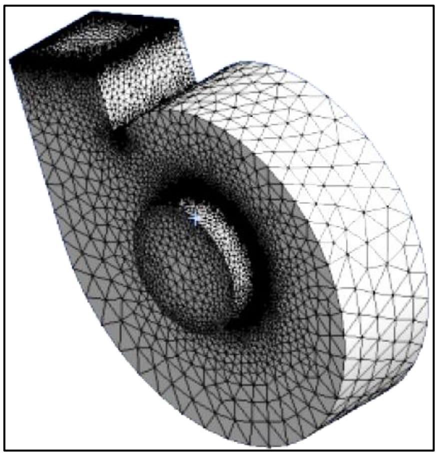

Fig. 2. Meshing of the radial fan

2.3 Governing Equations

The model considers an infinitesimally small fluid element moving with the flow. Although this element has a fixed mass; however, its shape and volume change as it moves downstream [17]. The governing equations of this simulation include the continuity and momentum equations. Because the mass is conserved, the time rate of change of the fluid element's mass is zero as it moves with the flow. The continuity equation is expressed as follows:

$$\frac{D\rho}{Dt} + \rho\nabla \cdot \mathrm{V} = 0 \tag{1}$$

The momentum equation in conservation form for x-direction is:

$$\frac{\partial(\rho u)}{\partial t} = \nabla \cdot (\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x \tag{2}$$

where ρ is the density, t is the time and V is velocity.

2.4 Boundary Conditions

Setting boundary conditions ensures that no data overstatement occurs. Table 1 lists the boundary conditions of the radial fan simulation analysis.

Table 1
Boundary conditions of the radial fan

Boundary condition	Value
Inlet	Atmospheric pressure (0)
Outlet	Static pressure (0)
Turbulence intensity	5%
Turbulence model	$k-\omega$

3. Results

3.1 Grid Independency Test (GIT)

The results for each case were converged using GIT, with the difference threshold set at 0.001 to ensure the convergence of the mass flow rate. The mesh was refined by increasing the number of iterations at the edges of the radial fan and by enhancing the face elements at the inlet and outlet. Tables 2, 3, and 4 present the GIT results for fans with 12, 16, and 20 blades, respectively.

Table 2
GIT results for 12 blade meshing

No	Node	Element	Mass flow rate (kg/s)	Difference
1	42431	223203	0.15554	-
2	57083	292785	0.15561	7×10^{-5}
3	95552	480102	0.15897	3.36×10^{-3}
4	100570	506945	0.15897	0

Table 3
GIT results for 16 blade meshing

No	Node	Element	Mass flow rate (kg/s)	Difference
1	57449	294131	0.21770	-
2	91936	460413	0.22092	3.22×10^{-3}
3	96349	483538	0.22092	0

Table 4
GIT results for 20 blade meshing

No	Node	Element	Mass flow rate (kg/s)	Difference
1	57449	294131	0.21770	-
2	91936	460413	0.22092	3.22×10^{-3}

3.2 Validation and Verification of Results

For validation, a previous experimental study by Aureliano *et al.,* [7] was referenced, and the validation results are shown in Figure 3. The percentage error between the current simulation and previous experimental results was 12.62%. This discrepancy may be due to the motor rotation speed efficiency in the experiment, whereas the simulation assumed a steady flow. The verification involved a numerical solution similar to that reported in [7], as illustrated in Figure 4. The percentage errors of the 12, 16, and 20 blades were 13.34%, 6.23%, and 1.92%, respectively (Table 5).

Although simulations should ideally match the exact geometry of the radial fan model, the precise geometry of the radial fan model has not been reported. Therefore, a similar fan with approximate geometry was modeled and simulated under identical conditions, resulting in an acceptable percentage error.

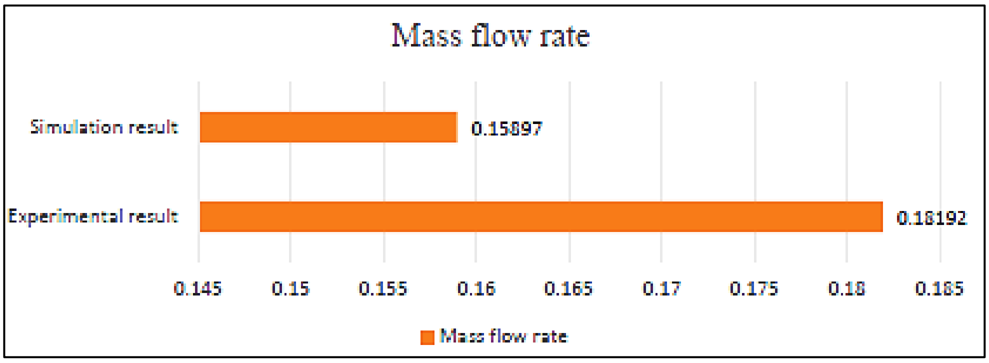

Fig. 3. Mass flow rates obtained from the simulation and experimental results

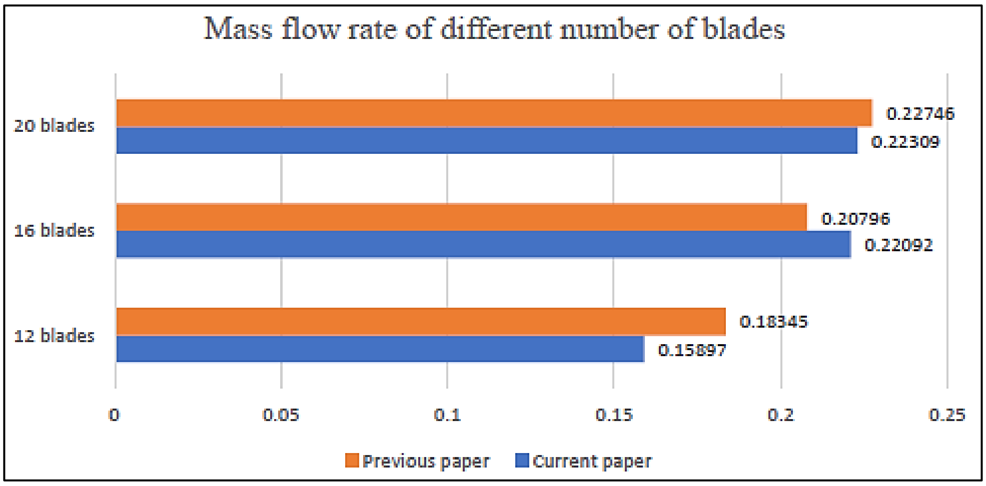

Fig. 4. Mass flow rate for different numbers of blades

Table 5
Percentage errors between previous and current papers

Blade number	Percentage error (%)
12	13.34
16	6.23
20	1.92

3.3 Velocity of the Radial Fan

Figure 5 shows the velocity streamlines at different blade counts. In the case of radial fans, the velocity plays a very significant role on the performance capacity of the fan because it determines the flow characteristics of the fan. The velocity distribution within the radial fan depends on the number of blades, rotation speed, and interaction of the fluids. When such a fluid flows into the fan through the inlet, the impeller, which is possessed of kinetic energy, moves it to and from the inlet via centrifugal force. This causes the fluid to flow at higher velocity in the radial direction through the blade passages of the turbine.

For fans with a larger number of blades, including 16 and 20 blade configurations, the fluid is retained within tight spaces between the blades. This resulted in an increase in the velocity of the flow since the same volumetric flow rate had to pass through the bladed section with a smaller distance between blades. For the 12-blade configuration, the blades were further apart from each other; as a result, the movement of the fluid was less limited, and the overall velocity of the fluid to be driven was significantly lower than that for the other blade configurations.

The velocity streamlines of radial fans with various blade numbers are characterized by this tendency. Figure 5 shows the velocity streamlines of the 12, 16, and 20 blades. From the above results, the velocity magnitude increased with increasing number of blades because of the reduction in the slip effect. More blades give the impeller a greater ability to transfer energy to the fluid, which increases the velocity of the fluid leaving the fan.

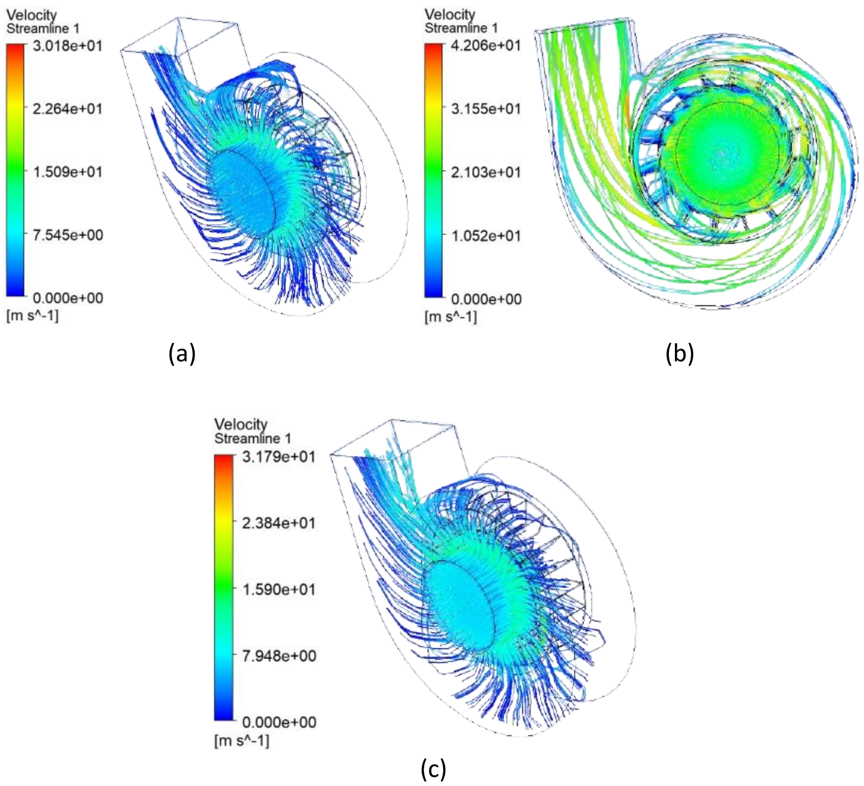

Fig. 5. Velocity streamline of a radial fan with different numbers of blades
(a) 12 blades (b) 16 blades (c) 20 blades

3.4 Mass Flow Rate

The simulation results obtained using ANSYS Fluent showed that the mass-flow rate increased with the number of blades, as shown in Figure 6. The number of blades influences both the performance and efficiency of any turbomachinery [18]. Thus, in fan design, the number of blades should be carefully set to optimize the fan performance. The optimal number of fan blades cannot be determined using theoretical relationships. It is often found through experimental studies. In general, the output flow is influenced by the number of blades. As the number of blades increased, the output flow of the radial fan increased [19]. As the blade number increases, the space between the blades decreases, which narrows the fluid path and reduces centrifugal effects; therefore, fluid flow increases [20].

Mainly, by increasing the number of blades, the characteristic curve moves upward. As mentioned above, the slip effect diminishes as the number of blades increases, and the impeller delivers more work to the fluid. Thus, at the same counter pressure, a fan with more blades achieves a greater flow than a fan with fewer blades.However, at the same flow rate, increasing the number of blades results in a larger difference in the pressure.

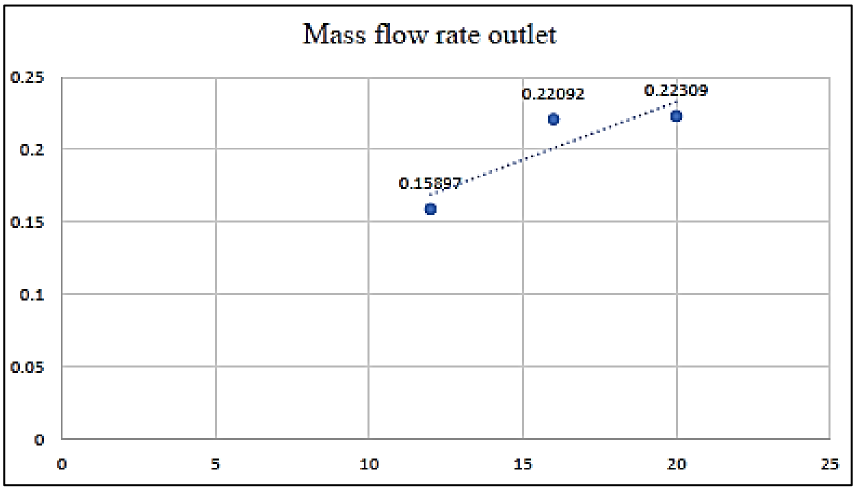

Fig. 6. Relationship between the mass flow rate and number of blades

4. Conclusions

In conclusion, the objectives of this study were successfully achieved. The first objective was to determine the flow characteristics near the blades of the radial fan. The second objective was to investigate the relationship between the number of blades and the fluid flow rate. Generally, the performance of a radial fan can be enhanced by adjusting the number of blades rather than altering the fan geometry. Increasing the blade count improved the performance in terms of the mass-flow rate at the outlet. Although blade number manipulation affects the mass-flow rate, the performances of different centrifugal fans should also be studied. In this study, examining the unsteady flow was more applicable to numerical analysis. Additionally, using the exact geometry in the experimental and simulation studies enhanced the data accuracy.

Acknowledgement
This research was supported by Universiti Tun Hussein Onn Malaysia (UTHM) through Tier 1 (vot Q449).

References
[1] Dick, Erik. "Fans." In *Fundamentals of Turbomachines*, p. 99-155. Cham: Springer International Publishing, 2022. https://doi.org/10.1007/978-3-030-93578-8_3
[2] Popoola, O. E., A. Taiwo, and J. A. Olaniran. "Performance evaluation of a developed centrifugal fan for chemical application in orchard farm." *Adeleke University Journal of Engineering and Technology* 4, no. 2 (2021): 20-27.
[3] Williams, A. E. *Industrial Energy Systems Handbook*. New York: River Publishers, 2022.
[4] Castorani, Vincenzo, Daniele Landi, Marco Mandolini, and Michele Germani. "Design optimization of customizable centrifugal industrial blowers for gas turbine power plants." *Computer-Aided Design and Applications* 16, no. 6 (2019). https://doi.org/10.14733/cadaps.2019.1098-1111
[5] Sanjosé, Marlène, Sandra Hub, Frieder Lörcher, and Stéphane Moreau. "Noise mechanisms in a radial fan without volute." In *Journal of Physics: Conference Series*, vol. 1909, no. 1, p. 012009. IOP Publishing, 2021. https://doi.org/10.1088/1742-6596/1909/1/012009
[6] Piwowarski, Marian, and Damian Jakowski. "Areas of fan research—A review of the literature in terms of improving operating efficiency and reducing noise emissions." *Energies* 16, no. 3 (2023): 1042. https://doi.org/10.3390/en16031042
[7] Aureliano, Filipe d'S., and Luiz Carlos V. Guedes. "Computational fluid dynamics (CFD): Behavioral study and optimization of the blades number of a radial fan." *Procedia Manufacturing* 38 (2019): 1324-1329. https://doi.org/10.1016/j.promfg.2020.01.157
[8] Galloni, E., P. Parisi, F. Marignetti, and G. Volpe. "CFD analyses of a radial fan for electric motor cooling." *Thermal Science and Engineering Progress* 8 (2018): 470-476. https://doi.org/10.1016/j.tsep.2018.10.003

[9] Ch, Santosh Varun, K. Anantharaman, and G. Rajasekaran. "Effect of blade number on the performance of centrifugal fan." *Materials Today: Proceedings* 72 (2023): 1143-1152. https://doi.org/10.1016/j.matpr.2022.09.185

[10] Ding, Hongchang, Tao Chang, and Fanyun Lin. "The influence of the blade outlet angle on the flow field and pressure pulsation in a centrifugal fan." *Processes* 8, no. 11 (2020): 1422. https://doi.org/10.3390/pr8111422

[11] Selvaraj, T., P. Hariharasakthisudhan, S. Pandiaraj, K. Sathickbasha, and Noorani Mohamed AB Aslam. "Optimizing the design parameters of radial tip centrifugal blower for dust test chamber application through numerical and statistical analysis." *FME Transactions* 48, no. 1 (2020): 236-245. https://doi.org/10.5937/fmet2001236S

[12] Zapata, Andrés, Carlos Amaris, Alexis Sagastume, and Andres Rodriguez. "CFD modelling of the ammonia vapour absorption in a tubular bubble absorber with NH3/LiNO3." *Case Studies in Thermal Engineering* 27 (2021): 101311. https://doi.org/10.1016/j.csite.2021.101311

[13] de Morais Cabral, Endyara, Josedite Saraiva de Souza, Hortência Luma Fernandes Magalhães, Túlio Rafael Nascimento Porto, Carlota Joaquina, Ricardo Soares Gomez, Wanderson Magno Paiva Barbosa de Lima, Elisiane Santana de Lima, and Antonio Gilson Barbosa de Lima. "Análise do processo de separação óleo-água em hidrociclones via CFD." *Research, Society and Development* 9, no. 11 (2020): e90991110610-e90991110610. https://doi.org/10.33448/rsd-v9i11.10610

[14] Franzke, Randi, Simone Sebben, Tore Bark, Emil Willeson, and Alexander Broniewicz. "Evaluation of the multiple reference frame approach for the modelling of an axial cooling fan." *Energies* 12, no. 15 (2019): 2934. https://doi.org/10.3390/en12152934

[15] Johansson, August, Benjamin Kehlet, Mats G. Larson, and Anders Logg. "Multimesh finite element methods: Solving PDEs on multiple intersecting meshes." *Computer Methods in Applied Mechanics and Engineering* 343 (2019): 672-689. https://doi.org/10.1016/j.cma.2018.09.009

[16] Cremonesi, Massimiliano, Alessandro Franci, Sergio Idelsohn, and Eugenio Oñate. "A state of the art review of the particle finite element method (PFEM)." *Archives of Computational Methods in Engineering* 27, no. 5 (2020): 1709-1735. https://doi.org/10.1007/s11831-020-09468-4

[17] Ferziger, Joel H., Milovan Perić, Robert L. Street, Joel H. Ferziger, Milovan Perić, and Robert L. Street. "Basic Concepts of Fluid Flow." *Computational Methods for Fluid Dynamics* (2020): 1-21. https://doi.org/10.1007/978-3-319-99693-6_1

[18] Li, Peng, Zhonghe Han, Xiaoqiang Jia, Zhongkai Mei, and Xu Han. "Analysis of the effects of blade installation angle and blade number on radial-inflow turbine stator flow performance." *Energies* 11, no. 9 (2018): 2258. https://doi.org/10.3390/en11092258

[19] Zhou, Shuiqing, Huaxin Zhou, Ke Yang, Haobing Dong, and Zengliang Gao. "Research on blade design method of multi-blade centrifugal fan for building efficient ventilation based on Hicks-Henne function." *Sustainable Energy Technologies and Assessments* 43 (2021): 100971. https://doi.org/10.1016/j.seta.2020.100971

[20] Si, Qiaorui, Asad Ali, Jianping Yuan, Ibra Fall, and Faisal Muhammad Yasin. "Flow-induced noises in a centrifugal pump: a review." *Science of Advanced Materials* 11, no. 7 (2019): 909-924. https://doi.org/10.1166/sam.2019.3617